International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 4 Issue 5, August 2020 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

# A CFD Study on Plate Heat Exchanger using Nanofluid with Different Corrugated Channels Configurations under Forced Convection Flow

**Madhuri Singh[1], Prof. Animesh Singhai[2]**

[1]Research Scholar, [2]Professor,

[1,2]Trinity Institute of Technology and Research, RGPV, Bhopal, Madhya Pradesh, India

## ABSTRACT

Notwithstanding the fact that various empirical experiments have been performed on heat transfer and pressure drop in the corrugated surface as a result of parametric variance of the individual surface pattern. Nonetheless, there are very unusual records of heat transfer properties and pressure rises across corrugated channels with various surface patterns. The purpose of this work is therefore to include a comparative analysis with various surface patterns such that researchers can establish an optimization tool to pick an effective solution. For this research, the thermal and hydraulic properties of the corrugated fluid channels proposed to be semi-circular and triangular Plate heat exchangers (PHE) was studied through computational fluid dynamics (CFD) simulations utilizing popular CFD software ANSYS CFX 17.0. The goal of this analysis was to numerically model and validate the semi-circular corrugated channel with dimensions / parameters taken from available literature, and then to model / analyze the triangular corrugated channel related to the triangular model. In fact, in order to promote trust in this model, a distinction was made between the expected values and the findings of the literature for exactly the same event. An entire fluid channel is simulated using nanoparticles volume fractions of $Al_2O_3$ and Reynolds number ranging from 0 to 6 % and 10,000–30,000. The results showed that compared to semi-circular model at Re 10000 with volume fraction 6% of $Al_2O_3$ in water, the Nusselt number for triangular model increases 6.23 % referred to semi-circular model. With progression with volume fraction, Nusselt number increases. For triangular channel with Re=10000, at 0% volume fraction Nu=129.16 while at 6% volume fraction Nu=159.35.

***KEYWORDS:*** *Corrugated Channel, CFD, Heat Exchanger, Heat Transfer Enhancement and Nusselt number*

***How to cite this paper**:* Madhuri Singh | Prof. Animesh Singhai "A CFD Study on Plate Heat Exchanger using Nanofluid with Different Corrugated Channels Configurations under Forced Convection Flow" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-4 | Issue-5, August 2020, pp.37-44, URL: www.ijtsrd.com/papers/ijtsrd31718.pdf

IJTSRD31718



## I. INTRODUCTION

The heat transfer is a very interesting field for economical, functional and environmental reasons; it is almost related to each aspect of human lives. Therefore, the enhancement of such field is quite essential. The importance of improving heat transfer performance is well known in the fields of industry and research. Heat transfer can be maximized by three main techniques, which are active, passive and compound techniques. All of the three techniques are needed recently in heat exchangers and many other applications to get a smaller size, effective, and low cost thermal transport devices. The passive methods do not require external power sources, such as special geometries, treated surfaces, extended surface (fins), rough surfaces, additives for fluid, and so on, whereas the active methods require an external power source (electrical/mechanical) to realize the advanced heat transfer mechanism, such as the stirring of the ferrofluid with an electromagnetic field, vibrating surface [1].

New techniques to improve heat transfer have recently appeared in engineering research, as the insert of nanoparticles or porous medium, with high thermal conductivity, in working fluids to increase their effectivity thermal conductivity and convective heat transfer coefficients [1, 2].

More advances in modern technology calls for the need to design processing equipment that complies with the economic, environmental, and energy saving with high efficiency of mass and heat transfers. Improving convection heat transfer is a very important topic for various forms of industrial and engineering applications and can be enhanced passively by modifying the flow structure and boundary conditions or by improving the thermo physical properties of the stream.

Corrugated structures are well known for heat transfer enhancement applied in various engineering problems such as heat exchangers, air-conditioning devices, refrigeration systems, chemical reactors, and fuel rod [1]. The high heat transfer rate and the lower pressure drop are the pivotal functional requirements for an efficient design, which are strongly influenced by the shape of the plate. Corrugated heat exchanger comprises of a stack of compact corrugated parallel plates preferably material with appropriate thermal conductivity. Corrugation increases turbulence to the fluid flowing over it that results in heat transfer enhancement particularly for gas to gas or gas to liquid flows.

The ability of a fluid medium to transfer a large amount of heat to a small temperature gradient increases the efficiency of energy conversion and improves the design and performance of heat exchangers. Research on product development in these networks has now been quite popular. Of this reason, utilizing nanofluids as cooling fluids in corrugated channels instead of conventional fluids will increase the thermal conductivity of base fluids and therefore further boost the thermal efficiency of heat exchangers with a more compact nature.

## II. LITERATURE REVIEW

Numerous works have been done on Plate Heat Exchangers (PHEs) and their thermal and hydraulic characterization data are available in the open literature. However, there is a wide discrepancy in these reported correlations and, prior to the start of this study, it was necessary to analyse the experimental facilities and procedures, the methods of data reduction, the results and the conclusions of some of the important works in the past.

**Melvinraj et.al (2014)** has investigated on a parallel flow heat exchanger corresponding ribbed tube heat exchanger has also been modeled and numerically analyzed. For designing and analysis purpose Pro-e and ANSYS 14.5 has been used respectively. The effectiveness of two heat exchangers has been compared using CFD. The ribbed heat exchanger effectiveness is more than that of simple heat exchanger. Due to the shape of ribbed helical tube fluid flow is not parallel but in swirls, which increases turbulence and thereby increasing the effectiveness [1].

**Kansal and Sahabat (2015)** deals with the study of shell and tube heat exchanger by using KERN method and CFD simulation. Main aim of this work is to determine effectiveness of shell and tube heat exchanger. Methanol has been used as a hot fluid in shell side its inlet temperature is 368 K whereas water was used as fluid flow in tube at the inlet temperature of 298 K. From simulation it has been found that outlet temperature of methanol is 313 K and 315.53 K from KERN method and CFD respectively. Whereas outlet temperature of water is 313 K and 308.43 K from KERN and CFD respectively. From KERN method the effectiveness of heat exchanger was found out to be 0.79 and from CFD it has been found to be 0.76. Both the results are in close agreement with each other [2].

**Hasanpour et al. (2016)** have experimentally studied a double pipe heat exchanger with inner tube corrugated filled with various categories of twisted tapes from conventional to modified types (perforated, V-cut and U-cut). The twist ratio, the hole diameter, the width and depth ratio of the cuts have been varied and the Reynolds number has been changed from 5000 to 15000. Overall more than 350 experiments were carried out. Nusselt number and friction factor for corrugated tube equipped with modified twist tapes are found out to be higher than typical tapes [3].

**Johnson et.al (2017)** studied the analytical design of the heat exchanger which has been also numerically analyzed. On the basis of standard k- ε modelling CFD analysis have been done. The solution of the problem yields when the optimum values of flow rate, outer diameter of pipe and inner diameter of pipe to be used at an effective length for a double pipe heat exchanger. When the stream processes for specified flow rates then it was treated for a given inlet to outlet temperature. From the result it has been found that the design and analysis of the double pipe heat exchanger would be a great success [4].

**R K Ajeel et.al (2017)** studied CFD study on turbulent forced convection flow of $Al_2O_3$-water nanofluid in semi-circular corrugated channel. Computational Fluid Dynamics (CFD) simulations of heat transfer and friction factor analysis in a turbulent flow regime in semi-circle corrugated channels with $Al_2O_3$-water nanofluid is presented. Simulations are carried out at Reynolds number range of 10000-30000, with nanoparticle volume fractions 0-6% and constant heat flux condition. The results for corrugated channels are examined and compared to those for straight channels. Results show that the Nusselt number increased with the increase of nanoparticle volume fraction and Reynolds number. The Nusselt number was found to increase as the nanoparticle diameter decreased. Maximum Nusselt number enhancement ratio 2.07 at Reynolds number 30,000 and volume fraction 6% [5].

**Junqi et al. (2018)** has experimentally investigated the thermal hydraulic characteristics for three types of fluids (R245fa, glycol & water) on plate heat exchanger surface. To overall evaluate the enhanced heat transfer, concept of pump power is provided. Using multiple regression method, dimensionless correlation equation of Nusselt number & friction factor are given. It is concluded that the plate chevron angle affect thermal hydraulic performance. Heat transfer increases with increase in chevron angle & vice versa [6].

**Sharif Asal et al. (2018)** used Computational Fluid Dynamics approach with the Reynolds stress model to investigate the influence of the apex angle on the thermal and hydraulic features of triangular cross-corrugated heat exchangers. The Reynolds number was varied from 310 to 2064. The numerical results varied by 5% than experimental results. On increasing the apex angle, pressure forces increase which lead to pressure drop along with heat exchanger coefficient. It is concluded that on increasing apex angle from $45^0$ to $150^0$, vorticity magnitude & pressure forces along the direction of flow increase which lead to higher heat transfer [7].

As summarized above, many researchers have investigated various aspects of corrugated channel geometry to enhance heat transfer. However, the heat transfer performances of many other corrugated shapes have not yet been reported.

The objectives of this study are:

A. To study the heat transfer performance of corrugated plate heat exchanger of different shapes.

B. To observe which configurations and parameters that gives the best results.

C. To study and modeling the heat transfer of corrugated plate heat exchanger using CFD simulation.

D. To simulate the flow and temperature fields in heat exchanger passages and to establish heat transfer and Nusselt number.

## III. COMPUTATIONAL MODEL

Figure 1. illustrates the simple structure of the current problem, it consists of two walls which are the upper and lower walls and the average distance between these walls (H) is 10 mm. The channel consists of three sections: flat-adiabatic, corrugated and planar-adiabatic sections. The planar portion length is $L_1$=400 mm (upstream segment), and $L_3$=100 mm (downstream segment). Upstream means flow at the edge of the lead while downstream means flow at the edge of the trailing. Corrugated wall length: $L_2$=200 mm (Test Section). The channel depth (W) is 50 mm. The upper and lower corrugated walls are heated isothermally while the flat walls are thermally insulated.

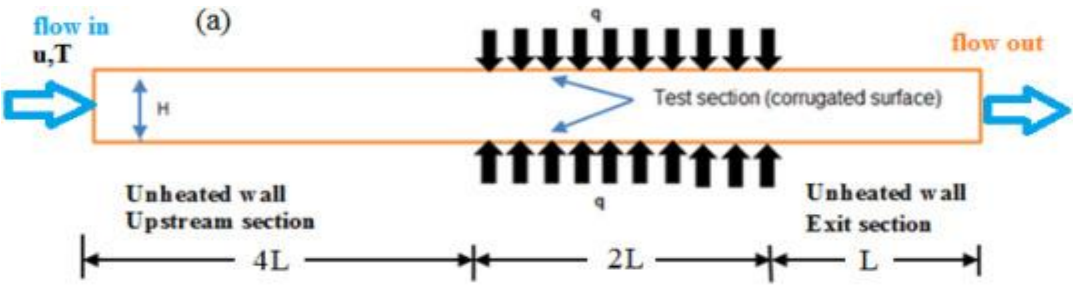


**Figure1. Physical model of the present study**

**Table1. Geometrical specification of corrugated channel**

| Corrugated Channel shape | Width(mm) | Pitch (mm) |
|---|---|---|
| Semi-circular | 5 | 15 |
| Triangular | 4.75 | 15 |

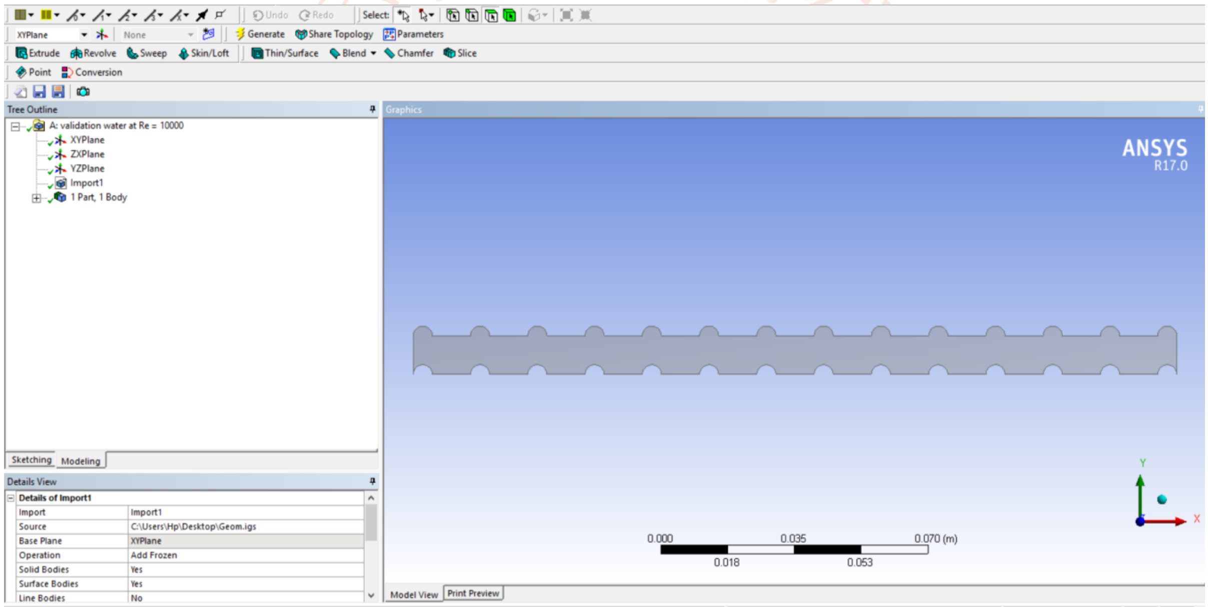

**Figure2. Semi-circular corrugated channel.**

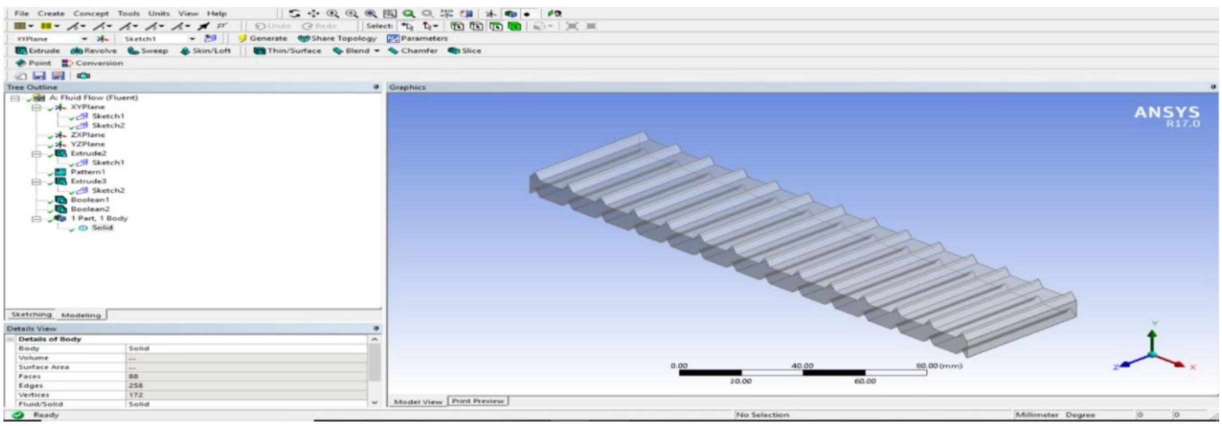

**Figure3. Triangular corrugated channel.**

A tetrahedron mesh is used for this purpose. The better grid on the middle part of the channel is selected along the wall for capturing of the wall impact and to save computing time. Underneath are the meshing of all geometry:

**Table2. Meshing Details**

| Corrugated Channel shape | Number of Nodes and Elements |
|---|---|
| Semi circular | 145640 and 105460 |
| Equilateral triangle | 129830 and 113550 |

## IV. NUMERICAL PROCEDURE

The nanofluid was used to check the properties of flow and thermal fields through computational simulation of a corrugated channel. The final volume method is employed by using the CFD commercial software ANSYS-FLUENT-V17.0 to resolve the governing equations with the relevant boundary conditions.
Pressure- velocity coupling – Scheme -SIMPLE
Pressure – Standard
Momentum – Second order
Energy - second order
Turbulent Kinetic Energy (k) – second order
Turbulent Dissipation Rate (e) - second order
For discretization of the equation least square cell based method is used.

As the cost of calculating every repetition increases and the solution more strongly depends on physical models, the k-ε model is a mediated and suitable simulation option. This layout also has a good streamlining curvature and is ideally adapted to the initial screening of alternate designs. It chooses the k-p turbulent model when the diffusion concept is approximated with the second order of upwind in momentum and energy equations.

In corrugated channels a turbulent flow-forced convection with a single step 3D model was carried out. The flow equations can be defined as:

**Continuity equation:**

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)=0$$

**U-momentum equation:**

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho uv)=-\frac{\partial P}{\partial x}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial u}{\partial y}\right]+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial u}{\partial x}-\frac{2}{3}\rho k\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial v}{\partial x}\right]$$

**V-momentum equation:**

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)=-\frac{\partial P}{\partial y}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial v}{\partial y}\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial v}{\partial y}-\frac{2}{3}\rho k\right]+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial u}{\partial y}\right]$$

**Energy equation:**

$$\frac{\partial}{\partial x}(\rho uT)+\frac{\partial}{\partial y}(\rho vT)=\frac{\partial}{\partial x}\left[\left(\frac{K}{c_p}+\frac{\mu_t}{Pr_t}\right)\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\frac{K}{c_p}+\frac{\mu_t}{Pr_t}\right)\frac{\partial T}{\partial y}\right]$$

**Table3. Nanoparticle and base fluid thermophysical characteristics of T=300 K**

| Material | Density ($Kg/m^3$) | Specific heat (J/Kg-K) | Thermal conductivity (W/m-K) | Dynamic viscosity ($N\text{-}s/m^2$) |
|---|---|---|---|---|
| Water | 998.2 | 4182 | 0.6 | 0.001 |
| $Al_2O_3$ | 3600 | 765 | 36 | ------- |

The effective characteristics of the nanofluid $Al_2O_3$ / water are described as follows:
The following are equations to evaluate density, thermal conductivity, and heat and nanofluid viscosity.

Nanofluid density and heat capacity are:

$$\rho_{nf}=\emptyset_p\rho_p+(1-\emptyset_p)\rho_{bf}$$
$$(\rho C_p)_{nf}=(1-\emptyset_p)(\rho C_p)_{bf}+\emptyset_p(\rho C_p)_p$$

In order to calculate the efficient thermal conductivity by using nanoparticles in corrugated channel, the impact of Brownian motion will be taken into consideration by using the empirical connection below:

$$K_{effective}=K_{static}+K_{Brownian}$$

$$K_{static} = K_{bf}\left\{\frac{K_p + 2K_{bf} - 2\emptyset_p(K_{bf} - K_p)}{K_p + 2K_{bf} + \emptyset_p(K_{bf} - K_p)}\right\}$$

$$K_{Brownian} = 5 \times 10^4 \beta\, \emptyset\, (\rho C_p)_{bf} \sqrt{\frac{kT}{\rho_p d_p}}\, f(T, \emptyset)$$

*Where,* Boltzmann constant, $k = 1.381 \times 10^{-23}\, J/K$

$$f(T, \emptyset) = (2.8217 \times 10^{-2}\, \emptyset + 3.917 \times 10^{-3})\frac{T}{T_0} + (-3.0669 \times 10^{-2}\, \emptyset - 3.9112 \times 10^{-3})$$

The effective dynamic viscosity of nanofluid is:

$$\mu_{nf} = \mu_{bf}\, A_1 e^{A_2 \emptyset}$$

*Where $A_1$=0.983 and $A_2$= 12.959*

**Table4. Distilled water and the $Al_2O_3$–water nanofluid with a different volume fraction at 300 K have thermophysical properties.**

| $\emptyset(\%)$ | $\rho(\frac{Kg}{m^3})$ | $c_p(\frac{J}{Kg-K})$ | $K(\frac{W}{m-K})$ | $\mu(\frac{N-s}{m^2})$ |
|---|---|---|---|---|
| 0 | 998.2 | 4182 | 0.6 | 0.001 |
| 2 | 1050.236 | 3947.74 | 0.6355 | 0.001273 |
| 4 | 1102.272 | 3735.60 | 0.6718 | 0.001650 |
| 6 | 1154.308 | 3542.59 | 0.7083 | 0.002139 |

The computer domain of the tested Channel has been subjected to boundary conditions. Consistent heat flow (q = 10 kW / m$^2$) was available, the walls were not slip-restricted, and the flat walls were thermally insulated. In addition, the inlet velocity and a temperature of 300 K have been adapted at the inlet, and the outlet pressure is used. The limit requirements for the dynamic flow can be seen as follows:

Input uses the limit condition "Velocity inlet" and uses the user's specified feature to determine the maximum formed velocity profile. The fluid velocity can be determined by the number of Reynolds. At the outlet of the channel is used the boundary condition of "pressure outlet" ($P_{out} = P_{atm}$), at which 1 atm is attached.

## V. RESULTS AND DISCUSSIONS

CFD computer modeling has been studied locally and globally with the influence of the geoms on the hydrodynamic and thermal activity of the fluxes that traverse the channels, with the flow phénomènes and heat transfer features for constant wall temperature. Next, in the case of semi-circular corrugated channels the temperature distribution inside the Reynolds scale ranges from 10000 to 30000. In addition, the Nusselt number was graphed and locally evaluated for the lower plate of the semi-circular corrugation tube. Subsequently for triangular corrugated channels were provided, identical to the semi-circular corrugated situation, the overall temperature distribution and the Nusselt number within the same variance of Reynolds number.

### 5.1. Validation of models of semi-circular corrugated channels use water as the working fluid

The Nusselt number values derived from the CFD simulation were contrasted with the values obtained from the R K Ajeel et.al (2017) analysis [5]:

**Table5. displays the values of the Nusselt number determined from the CFD simulation relative to the values derived from the study carried out by R K Ajeel et.al (2017)[5] for the corrugated semicircle channel utilizing water as a working fluid.**

| S. No. | Reynold's Number | Nusselt Number ( Base Paper) | Nusselt Number (Present Study) |
|---|---|---|---|
| 1. | 10000 | 124 | 124.12 |
| 2. | 15000 | 183 | 183.18 |
| 3. | 20000 | 239 | 241.41 |
| 4. | 25000 | 288 | 290.50 |
| 5. | 30000 | 348 | 348.08 |

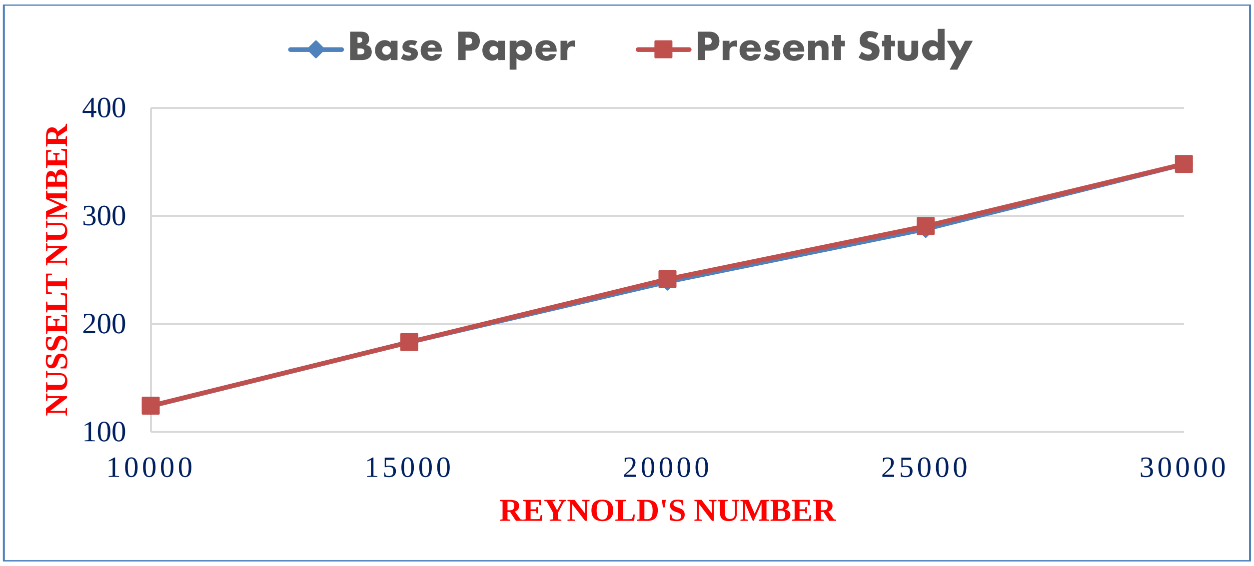


**Figure4. Comparison of Nusselt number calculated Present study with base paper for semicircle corrugated channel.**

It is noted from the aforementioned analysis that the significance of the Nusselt number derived from the CFD study is similar to that of the Nusselt number obtained from the base text. And we may assume the corrugated channel CFD pattern is right here.

### 5.2. Evidence for Corrugated Triangular Channel

At triangular corrugated tube, the turbulent induced convection of $Al_2O_3$-water nanofluid was studied. The CFD findings obtained were shown in terms of Nusselt number due to the influence of the various models, ϕ and Reynold's amount.

In this, the 3-D turbulent forced convective flow of $Al_2O_3$ – water nanofluid for triangular corrugated channels over Re ranging from 10,000 to 30,000 was examined under constant flux of heat, whereas ϕ ranged from 0 to 0.06.

**Table6. Nusselt number for Triangular corrugated channel at different Reynold's number and $\phi$ ranged from 0 to 0.06**

| Reynold's number | Nusselt number | | | |
|---|---|---|---|---|
| | $\phi$ =0 % | $\phi$ =2% | $\phi$ =4 % | $\phi$ =6 % |
| 10000 | 129.16 | 142.97 | 158.88 | 159.35 |
| 15000 | 190.88 | 200.27 | 220.96 | 227.04 |
| 20000 | 263.24 | 259.53 | 270.7 | 307.63 |
| 30000 | 360.03 | 366.86 | 398.34 | 438.3 |

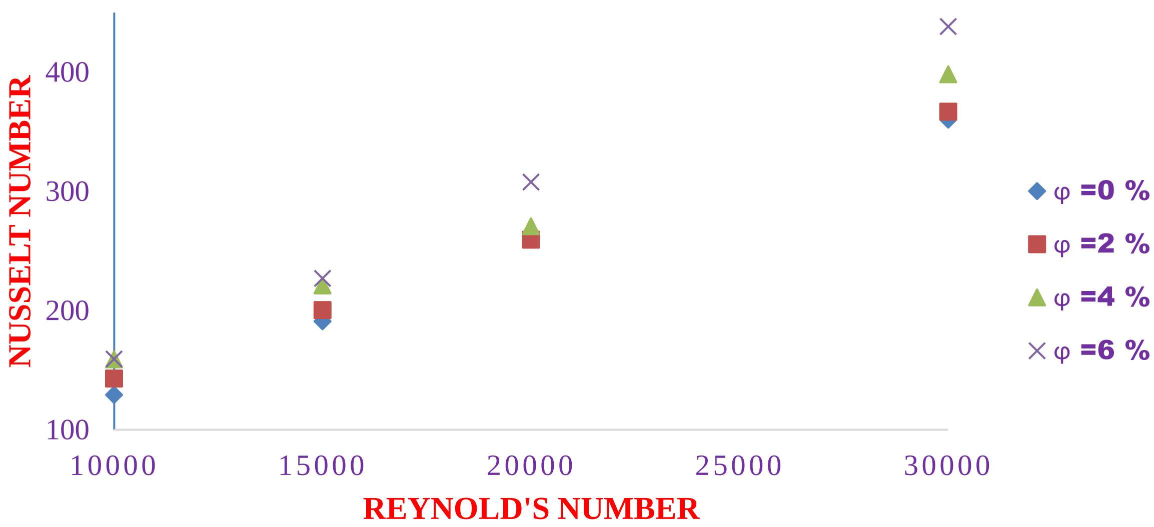


**Figure5. Nusselt number for Triangular corrugated channel at different Reynold's number.**

### 5.3. Comparison of Nusselt number calculated through CFD analysis for different corrugated channel

**Table7. Comparison of Nusselt number for different corrugated channel**

| Reynold's number | Corrugated Shape | Nusselt number | | | |
|---|---|---|---|---|---|
| | | $\phi$ =0 % | $\phi$ =2% | $\phi$ =4 % | $\phi$ =6 % |
| 10,000 | Semi-Circular | 125 | 130 | 140 | 150 |
| | Triangular | 129.16 | 142.97 | 158.88 | 159.35 |
| 15,000 | Semi-Circular | 183 | 190 | 200 | 218 |
| | Triangular | 190.88 | 200.27 | 220.96 | 227.04 |
| 20,000 | Semi-Circular | 240 | 248 | 252 | 290 |
| | Triangular | 263.24 | 259.53 | 270.7 | 307.63 |
| 30,000 | Semi-Circular | 348 | 355 | 380 | 425 |
| | Triangular | 360.03 | 366.86 | 398.34 | 438.3 |

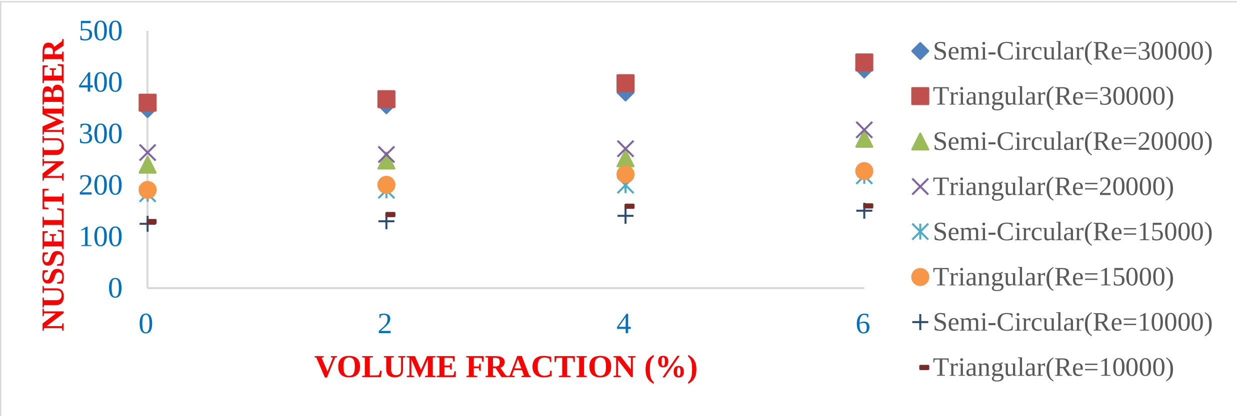


**Figure6. Comparison of Nusselt number for different corrugated channel.**

## VI. CONCLUSIONS

The present research explores the capacity of a commercial CFD code to predict the characteristics of flow and heat transfer in a narrow channel of corrugated roof, having a particular corrugated shape. Using a CFD code allows computing for different geometric configurations to evaluate and study their effects closely. In this way the engineer will improve the output for a specified geometry (i.e., increase the ratio of heat transfer to friction losses). In this study, a numerical simulation of a corrugated channel with two corrugation profiles is performed on the thermal performance comparison. Semicircle and triangular shapes are handled as corrugation profiles for corrugated channel walls using $Al_2O_3$ and Reynolds volume fractions varying from 0 to 6 percent and 10,000 to 30,000 respectively. The conclusions from the test are as follows:

- Compared to the Re 10000 semi-circular model of 6% volume fraction of Al2O3 in water, the Nusselt number for the triangular model rises 6.23% for the semi-circular model.
- The heat transfer rate increases sensibly as the water flows through the corrugated channel with progression with the Reynolds number. This can be calculated by growing the Nusselt number for the triangular corrugated channel at Re 10000 from 129.16 to 360.3 at Re 30000.
- As the $Al_2O_3$ –water nanofluid flows through the corrugated channel, with progression with volume fraction, Nusselt number increases. For triangular channel with Re=10000, at 0% volume fraction Nu=129.16 while at 6% volume fraction Nu=159.35.
- The flow vortices are formed as the Al2O3 -water nanofluid passes through the corrugated pipe. The vortex gets bigger and as Reynolds number increases the core shifts downstream. Within the periodic corrugations, the vortices generation is due to the pressure gradient which causes the reduction of the flow velocity. For triangular model, the largest vortices are seen, whereas for semi-circular model, the smallest are visualized.

It is assumed, above all, that all hydrodynamic and thermal activities are purely important for the amount of Reynolds and corrugation patterns.

## REFERENCES


[1] R Melvinraj C, C Vishal Varghese, Wilson Vicky, Thazhatha Thomas Jerry, Kakkasserymithunk, Simon J, 2014, "Comparative Study Of Heat Exchangers Using CFD, Int. Journal Of Engineering Research And Applications" Volume 4, Issue 5, PP-118-120.

[2] Kansal Santosh, Fatehmohd. Shabahat, 2015, " Design And Performance Evaluation Of Shell And Tube Heat Exchanger Using CFD Simulation", International Journal Of Research And Technology, Volume 3,Issue 7, PP-1663-1666.

[3] Hasanpour A., M. Farhadi, and K. Sedighi. "Experimental heat transfer and pressure drop study on typical, perforated, V-cut and U-cut twisted tapes in a helically corrugated heat exchanger." International Communications in Heat and Mass Transfer 2016; 70; 126-136.5

[4] Johnson G., Aikaterini A. Mouza, and Spiros V. Paras. "Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code." Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology 2017; 29(8): 923-930.

[5] R K Ajeel and W S I W Salim1, 2017, " A CFD study on turbulent forced convection flow of $Al_2O_3$- water nanofluid in semi-circular corrugated channel", IOP Conf. Series: Materials Science and Engineering 243 (2017) 012020.

[6] Junqi, Dong, Zhang Xianhui, and Wang Jianzhang. "Experimental Study on Thermal Hydraulic Performance of Plate-Type Heat Exchanger Applied in Engine Waste Heat Recovery." Arabian Journal for Science and Engineering 2018; 43(3): 1153-1163.

[7] Sharif Asal, Bernd Ameel, Ilya T'Jollyn, et al. "Comparative performance assessment of plate heat exchangers with triangular corrugation." Applied Thermal Engineering 2018.

[8] Abed Waleed Mohammed, Ahmed Mohammed Abed, 2008," Numerical Study Of Laminar Forced Convection Heat Transfer And Fluid Flow Characteristics In A Corrugated Channel", Journal Of Engineering And Development, Volume 14, Issue 3, PP 70-85.

[9] Goodarzi Marjan, Amiri Ahmad, Goodarzi Mohammad Shahab, Safaei Mohammad Reza, Karimipour Arash, Languri Ehsan Mohseni, Dahari Mahidzal, 2008, "Investigation Of Heat Transfer And Pressure Drop Of A Counter Flow Corrugated Plate Heat Exchanger Using MWCNT Based Nanofluids", International Communication In Heat And Transfer, PP 172-179.